FLUID STRUCTURE INTERACTION IN MULTIPHASE MIXING VESSEL

Matej Vesenjak, Zoran Ren, Matjaž Hriberšek
University of Maribor, Faculty of Mechanical Engineering,
Smetanova 17, SI-2000 Maribor, Slovenia

Summary In this paper a problem of fluid-structure interaction in a mixing vessel is solved with weakly coupled fluid-structure interaction computational analysis. The computational fluid dynamics code CFX, based on the finite volume method, is used for determination of the multiphase flow field (water and air) in the mixing vessel. The results in form of pressure distribution are then applied to the blade model, which is then analysed with the finite element structural analysis system MSC.visualNastran for Windows. The resulting deformations and stresses are evaluated. The proposed procedure can be effectively used for optimization of structures with significant fluid flow influences.

INTRODUCTION

Two or more physical systems frequently interact with each other, where the independent solution of one system is impossible without a simultaneous solution of the others. Such systems are known as coupled, where coupling may be weak or strong, depending on the degree of interaction. In case of weakly coupled systems, where one physical system has only a small influence on other systems, only one-way information stream is sufficient. Strongly coupled systems necessitate a full exchange of information between two or several interacting physical systems. An obvious coupled system is that of a dynamic fluid-structure interaction. [8]

In this paper a computational analysis of the fluid-structure interaction in a mixing vessel is presented. In mixing vessels the fluid can have a significant influence on the deformation of blades during mixing, depending on speed of mixing blades and fluid viscosity. For this purpose a computational weakly coupled analysis has been performed to determine the multiphase fluid influences on the mixing vessel structure. The multiphase fluid field in the mixing vessel was first analyzed with the computational fluid dynamics (CFD) code CFX. The results in the form of pressure were then applied to the blade model, which was the analysed with the structural code MSC.visualNastran for Windows, which is based on the finite element method (FEM).

CFD ANALYSIS OF THE VESSEL

In this paper the mixing of two fluids (water and air) in a mixing vessel is considered. To perform the CFD analysis of the posed physical problem, the mixing tank vessel containing four baffles is discretised with finite volumes. In the interior four rotating impeller blades are connected to a shaft which runs vertically through the vessel (Figure 1). Air is injected into the vessel through an inlet pipe located bellow the impeller blade at a speed of 5 m/s. The angular velocity of the shaft is 3 s^{-1}. The double symmetry of the vessel allows for only ¼ of the vessel to be modelled. The model was analyzed with the finite volumes CFD code CFX and the results in the form of pressure filed on the blade are shown in Figure 2. In spite of the fact that the most important values are those on a boundary between the fluid and the structure, the results of the analysis can be evaluated in every single point of the fluid. The most important results are the loads of the fluid applied to the vessel structure. It was determined that the highest pressure is located at the upper corner of the blade, where the blade is fixed to the shaft. It must be taken into consideration that the results are always conservative because the CFX code treats the fluid boundaries (in our case vessel and blades) as rigid. The computed forces (pressure) were then being used as the boundary conditions in the structural analysis of the vessel blade.

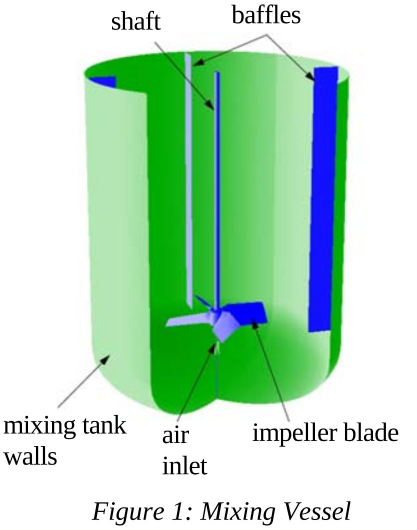

Figure 1: Mixing Vessel

TRANSFORMATION OF CFD DATA TO NASTRAN

The first condition that must be fulfilled is to establish an equilibrium state between the fluid and the structure. The fluid pressure (force) which acts on the structural surface must be equal to the force in the structure, which acts in the opposite direction

$$\mathbf{p} = p \cdot \mathbf{n}_{\text{structure}} = p \cdot \mathbf{n}_{\text{fluid}} \,, \qquad (1)$$

where $\mathbf{n}_{\text{structure}} = -\mathbf{n}_{\text{fluid}}$ are the external interface normals between the fluid and the structure. To transfer pressure loads from the CFD mesh to the FEM mesh, it is most convenient if the nodes on interfaces are coincident, since the data transfer is the trivial. In case where those nodes are not coincident, the loads must be properly interpolated (the load field must not change during data transfer) in order to achieve reasonable results. In case of a weakly coupled analysis the constraints are defined with regard to the physical constraints since they are not a result of the computational fluid dynamics. They are defined in the displacement vector of a finite element.

The CFX has the ability to export results in the ASCII format. In case of the presented coupled analysis the most important exported data were: nodes positions (used for establishing a new finite element mesh of the blade model) as well as water and air pressures acting on the blade. The results from the CFX and the finite element mesh data were then written in a proper MSC.visualNastran for Windows form.

The first step was the definition of node positions for the structural analysis. The same node positions as used in the CFD analysis were transferred, although only triangular elements were used. After defining the nodes, loads from both fluids (water and air) were summarized and redefined. Because the CFX is not able to export the topology of elements, the finite element mesh was modelled manually according to the transferred nodes.

FEM ANALYSIS OF THE BLADE

The finite element method was used for determination of the deformation and stress fields of the blade under the influence of computed fluid forces shown on Figure 2.

For the structural analysis of the blade the transformed data, described in the previous chapter, were imported into the MSC.visualNastran for Windows system. Then the material properties of the blade were defined as follows: elasticity modulus 210000 MPa and Poisson ration 0,3. The blade has a constant thickness of 35 mm and was meshed with 3-node shell elements. It was fixed at the edge were it is welded to the shaft (right edge of the discretised model shown on Figure 3).

A linear elastic quasi-static computational analysis resulted in blade deformations and stresses as shown in Figure 3. The largest computed deformation of the blade was 0.076 mm at the leftmost free corner, while the stresses were the highest at the point of attachment to the shaft in the upper right corner and equal to 32.23 MPa.

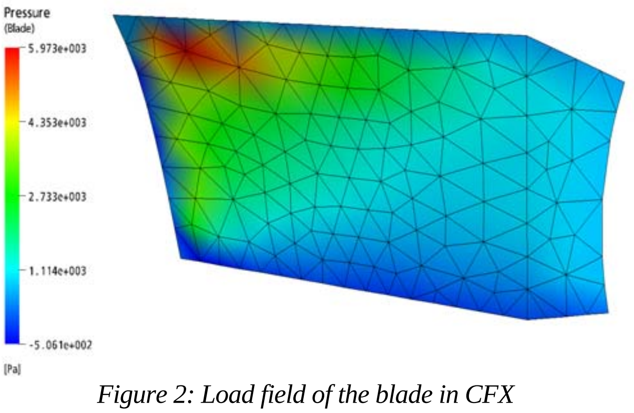

Figure 2: Load field of the blade in CFX

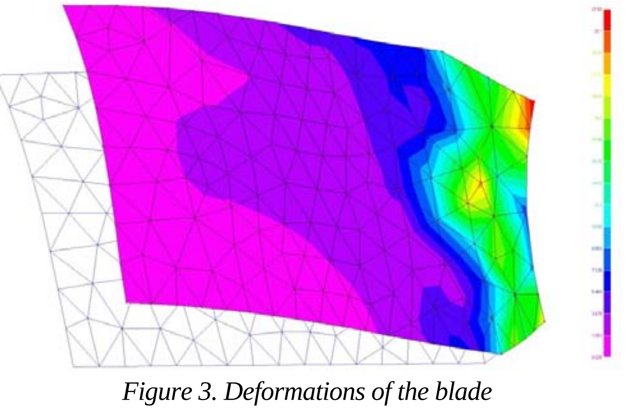

Figure 3. Deformations of the blade

CONCLUSION

The deformations of the blade in a multiphase mixing vessel were determined by means of a weakly coupled computational analysis. The fluid field was analysed with the computational fluid dynamics code CFX. The results in the form of the pressure (forces) were then used for structural analysis of the blade with the finite element system MSC.visualNastran for Windows.

Described procedure can be applied to any problem concerning a weakly coupled fluid-structure interaction. However, a significant research effort is directed to develop robust and accurate fully coupled fluid-structure interaction analysis methods, also considering material and geometrical nonlinearities.

References

[1] CFX Manual, Tutorial
[2] Kuzmin, D.: Introduction to Computational Fluid Dynamics. University of Dortmund, Dortmund.
[3] Kilroy K.: MSC/Nastran – Quick Reference Guide. The MacNeal-Schwendler Corporation, 1998.
[4] NAFEMS: A Finite Element Primer. NAFEMS. Glasgow, 1992.
[5] Schäfer M., Teschauer I., Kadinski L., Selder, M.: A numerical approach for the solution of coupled fluid – solid and thermal stress problems in crystal growth processes. Elsevier, Darmstadt, 2002.
[6] Stein K. R.: Simulation and Modeling Techniques for Parachute Fluid – Structure Interaction. Doctor Thesis, University of Minnesota, Minnesota, 1999.
[7] The fluid equations. [Online] http://astron.barkeley.edu/~jrg/ay202/node152.html [8. September. 2003].
[8] Zienkiewicz O. C., Taylor R.L.: The Finite Element Method – Volume 1 – The Basis. McGraw-Hill Ltd., London, 2000.